RESEARCH

23(79), September 1, 2014

ISSN 2278–5469 EISSN 2278–5450

Discovery

Combustion instabilities in solid rocket motor using PDF combustion

Muneshwar S☼, Asokan R

School of Aeronautical Sciences, Hindustan University, Chennai-603103, India

☼**Corresponding author:**
S. Muneshwar, School of Aeronautical Sciences, Hindustan University, Chennai-603103, India: Email: skmtifac@gmail.com

Publication History
Received: 12 July 2014
Accepted: 27 August 2014
Published: 1 September 2014

Citation
Muneshwar S, Asokan R. Combustion instabilities in solid rocket motor using PDF combustion. *Discovery*, 2014, 23(79), 137-143

ABSRTACT
The present work has been done numerically to simulate the combustion instabilities in solid rocket motors. Hot flow with the flow of oxidizer and fuel injection is considered in this investigation. The flow is assumed to be steady in the numerical simulations. It must be clearly understood that the flow-acoustic coupling which is of primary interest in the context of rocket motor instabilities cannot be simulated entirely by incompressible flow calculations alone. Numerical calculations have been carried out to determine the combustion instability, temperature, species and specific heat with the effect of varying the velocity. The combination of the above entities in the Solid Rocket Motor causes some severe damage in the combustion chamber. By identifying the source, the damage can be minimized by altering the design and strengthening the structures at the required spot.

Keywords: Solid rocket, combustion, combustion instability, combustion chamber.

1. INTRODUCTION
An important property of a combustion chamber is combustion stability. To have combustion stability, the flame must remain stable at varying fuel mixtures, inlet temperatures, turbulence levels, flow speeds and so on. The two limits mainly depend on the mass flow of air passing through the combustion chamber. Flames have trouble surviving at high flow velocities. A high flow velocity is, of

course, linked to a high mass flow. Raheem, (2004) has done numerical calculations of the unsteady flow in a scale model rocket motor chamber experimentally studied by Shanboughe et al. (2003). The main objective of this work was to find out how well (if at all) the frequencies of the oscillations can be predicted by numerical simulations. The flow is assumed to be incompressible, and axi-symmetric. All the results reported here are second order accurate in space and time. FFT of the pressure-time data has been performed to extract the dominant frequencies of the oscillations. Values for the dominant frequencies (as well as the second dominant one, in some cases) obtained from the numerical calculations are compared with the experimental values. Sanjeev, (2004) made an investigation of combustion instability in solid rocket motors was conducted using perturbation techniques, with particular emphasis placed upon understanding the fluid dynamics of the chamber environment.

Table 1

Dimension for model geometry in X-Y direction

Decoupler Dimensions			**Port Dimensions**		**Downstream part Dimensions**				
D1 in mm	**D_2 in mm**	**L_1 in mm**	**L_2 in mm**	**D_3 in mm**	**L_3 in mm**	**L_4 in mm**	**L_5 in mm**	**D_4 in mm**	**D_5 in mm**
25	15	15	150	5	14.9663	29	35	31	10

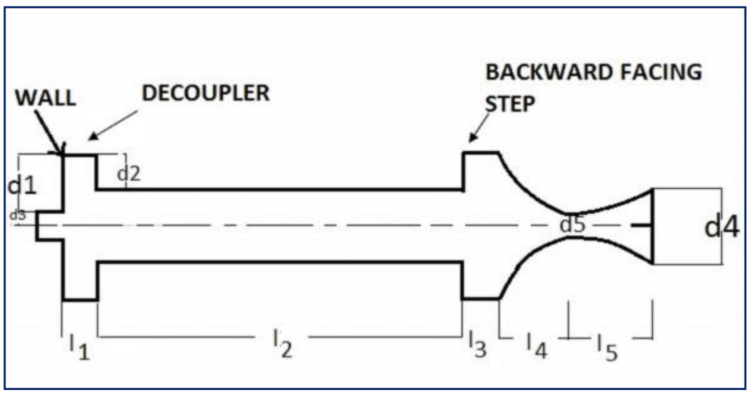

Figure 1

Scale model of a Solid Rocket Motor

Kulhanek, (2012) had done the design, analysis and simulation of the Rocket propulsion system. This paper details the functionality of a software program used to streamline a rocket propulsion system design, analysis and simulation effort. The program aids in unifying the nozzle, chamber and injector portions of a rocket propulsion system design effort quickly and efficiently using a streamlined graphical user interface (GUI).

Pandey, (2010) had done CFD analysis of pressure and temperature for a rocket nozzle with four inlets at Mach 2.1 is analyzed with the help of fluent software. When the fuel and air enter in the combustion chamber according to the x and y plot, it is burning due to high velocity and temperature and then temperature increases rapidly in combustion chamber and convergent part of the nozzle. Vipul, (2011) had worked on the prediction of temperature profile inside the rocket nozzle and using temperature profile prediction the velocity at the exit of nozzle. The prediction code generated is used in outer-atmospheric conditions with variation of pressure in vacuum consideration in outer-atmospheric condition. Fluent and Gambit software are used for CFD analysis in the nozzle. The author done research for predicting and quantifying undesirable transient axial combustion instability symptoms necessitates a comprehensive numerical model for internal ballistic simulation under dynamic flow and combustion conditions. A numerical model incorporating pertinent elements, such as a representative transient, frequency-dependent combustion response to pressure wave activity above the burning propellant surface, is applied to the investigation of scale effects (motor size, i.e., grain length and internal port diameter) on influencing instability-related behavior in a cylindrical-grain motor. In the present work the model of the Solid Rocket Motor is taken from the experimental model of Shanbhogue et al, (2003). The model is the scale model of the same. The model and the dimensions are shown in figure 1 and table 1. In this project, the scale model provided above is designed in GAMBIT and the meshing is done and the pressure instabilities are calculated and numerically studied using the software called FLUENT. The objective of this project is to investigate the frequencies of the oscillations in combustion chamber of solid rocket motor.

2. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITION

The computational domain is drawn roughly and the boundary conditions are assigned for the scale model (Figure 2). Any solution of a set of PDE's requires a set of boundary condition. The boundary conditions are required at all boundaries that surround the flow domain. Also, in this computational work velocity has been used as an inlet parameter. Outflow boundary condition has been used to model flow exists where details of the flow velocity and pressure are unknown. The value of all the variables on this boundary has obtained by extrapolation from the interior. Wall boundary conditions have been used to model the boundary conditions at the wall of the domain. These also used to bound fluid and solid. No-slip condition has been imposed on the wall. This

can be used to model a momentum source and it is possible to specify the shear stress. Symmetry boundary condition has been used when the physical geometry and flow field has mirror symmetry and at the symmetry plane the parameters like, Zero normal velocity and zero normal gradients at all variables have been satisfied.

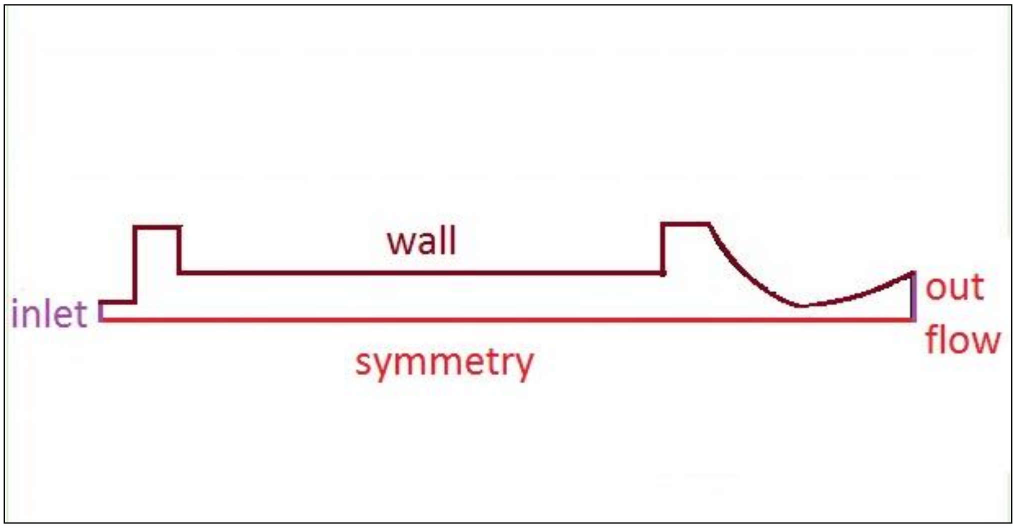

Figure 2

Computational domains with boundary conditions

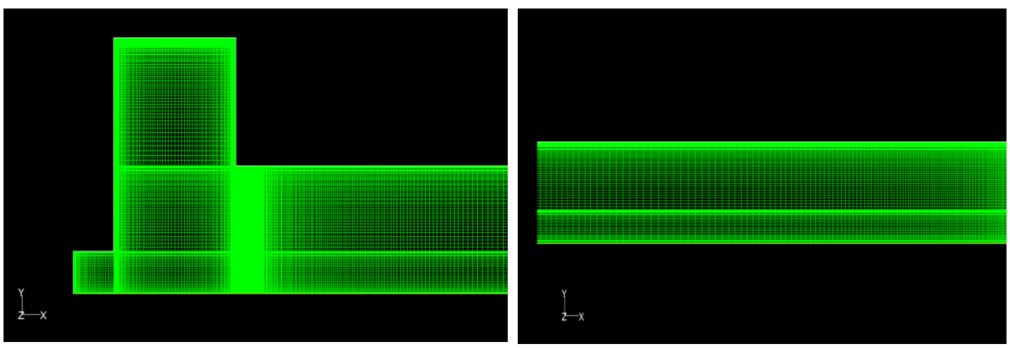

Figure 3

Pattern of meshing near the inlet and the wall

2.1. Solver Setting and Solution Methodology

A combustion instability model of SRM has been considered with reference to experimental observation. The simulation has been done for obtaining flow field results which was not possible through experimental observation of the researchers. Using fluent software, the simulation has been carried out. Implementing the experimental condition, the cold flow condition has been considered. The maximum velocity of the cold flow has been kept as 75m/s which is much less compared to sound velocity in air at ambient condition (300K), and flow has been considered as an incompressible. Also, the assumption of incompressible has been justified in the context to Mach number, which is well below than 0.3. The flow has been studied by using incompressible Navier-Stokes equation. The computational domain with boundary conditions has been presented. Based on the experimental condition, the average velocity has been selected at the inlet. Also, the turbulent intensity has been used for the turbulent case study, and the turbulent intensity for the flow field has been selected based on the relationship,

$$I = \frac{u'}{u_{avg}} = 0.16(\mathrm{Re})^{-1/8}$$

This is the empirical correlation for fully developed pipe flow. Since, FLUENT uses a control volume based technique to convert the governing equations of the flow into algebraic equations, and solved numerically. So the computational domain has been discretized using quadrilateral and triangular unstructured meshes. Using the Reynolds-Averaged Navier- Stokes (RANS) equations, which is used by finite volume method, the current problem has been solved. For practical configurations, such full scale solid rocket motors, Direct Numerical Simulation (DNS) and Large eddy Simulation (LES) cannot be used, because it is quite expensive and generates unsuitable resolution. Therefore, the use of Reynolds-Average Navier-stokes equations containing turbulence model is currently the viable way for predicting the mean and turbulent quantities of complex flow. Initially, steady state calculations are carried out to remove the transient till the residuals of all governing equations are of the order of 10^{-5} and then unsteady calculation has been done. The segregated implicit solver along with the SIMPLE scheme has been used. The realizable $\varepsilon\ \kappa -$ model has been used to model turbulence effects, because it has been found to give good and reliable predictions over the entire range of the flow rates and differs from the standard $\varepsilon - k$ model in that it contains a new formulation for

the turbulent viscosity and a new transport equation for the dissipation rate, ε based on the dynamic equation for the transport of the mean square vorticity fluctuation. The meshing is such a way that the meshing is finer in the inlet and throughout the path of the inlet. The meshing is finer near the wall (Figure 3). The meshing is finer near the wall and less fine as they descend towards the center. The following figure shows the grid for the wall.

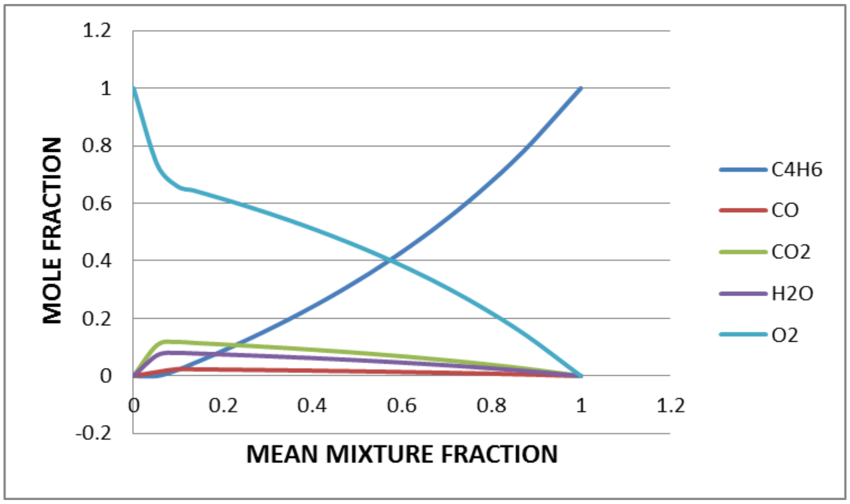

Figure 4

Graph for Mean mixture fraction and mole fraction for species

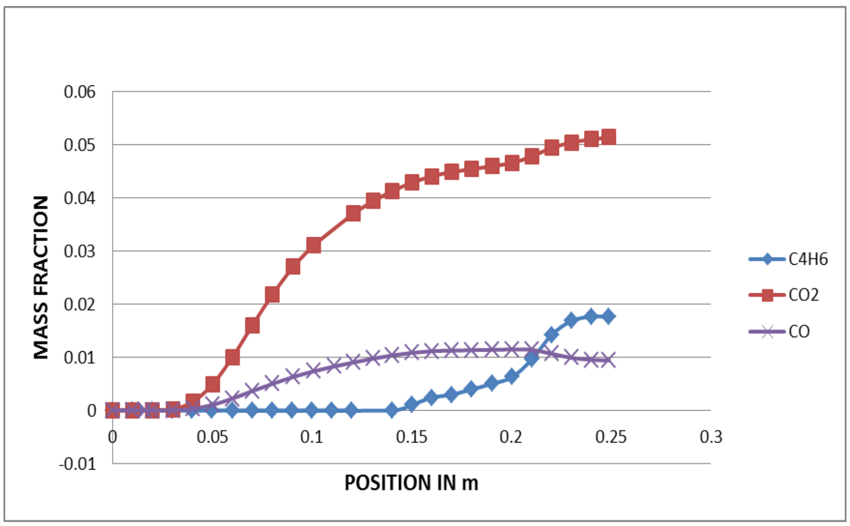

Figure 5

Mass fraction along with position

2.2. Non-Premixed Combustion Model

In order to simulate the Solid Rocket Motor combustion the Non-Premixed Combustion Model approach is used in FLUENT. In this approach individual species transport equations are not solved. Instead, transport equations for one or two conserved scalars (the mixture fractions) are solved and individual component concentrations are derived from the predicted mixture fraction distribution. This approach has been specifically developed for the simulation of turbulent diffusion flames and offers many benefits over the finite-rate formulation. In the conserved scalar approach, turbulence effects are accounted for with the help of a probability density function or PDF. The reacting system is treated using a flame sheet (mixed-is-burned) approach or chemical equilibrium calculations. In this non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. This is in contrast to premixed systems, in which reactants are mixed at the molecular level before burning. Examples of non-premixed combustion include methane combustion, pulverized coal furnaces, and diesel (compression) internal-combustion engines.

Under certain assumptions, the thermochemistry can be reduced to a single parameter: the mixture fraction. The mixture fraction, denoted by *f*, is the mass fraction that originated from the fuel stream. In other words, it is the local mass fraction of burnt and unburnt fuel stream elements (C, H, etc.) in all the species (CO_2, H_2O, O_2, etc.). The approach is elegant because atomic elements are conserved in chemical reactions. In turn, the mixture fraction is a conserved scalar quantity, and therefore its governing transport equation does not have a source term. Combustion is simplified to a mixing problem, and the difficulties associated with closing non-linear mean reaction rates are avoided. Once mixed, the chemistry can be modeled as in chemical equilibrium, or near chemical equilibrium with the laminar flame let model.

2.3. Equilibrium Mixture Fraction/ PDF Model

The non-premixed modeling approach involves the solution of transport equations for one or two conserved scalars (the mixture fractions). Equations for individual species are not solved. Instead, species concentrations are derived from the predicted mixture fraction fields. The thermochemistry calculations are preprocessed in **prePDF** and tabulated for look-up in **FLUENT**. Interaction of turbulence and chemistry is accounted for with a probability density function (PDF).

In this model the inlet velocity is given as 50 m/s. The wall has a constant temperature of 1200 K the inlet temperature is assumed as 1500 K and the outlet temperature is 2000 K. The Reynolds number based on the inlet dimension and the average inlet

velocity is about 100,000. Thus, the flow is turbulent. Only half of the domain width is modeled because of symmetry. At first the steady flow is considered. The non-adiabatic system is considered. In order to simulate Non-Premixed Combustion, the PDF is to be calculated. In this PDF the process is considered as non-adiabatic. Before that, the viscous model is selected as k-epsilon (two equations). All the turbulent calculations have been performed using the "realizable" ε − k model. The realizable ε κ − model has been used to model turbulence effects, because it has been found to give good and reliable predictions over the entire range of the flow rates.

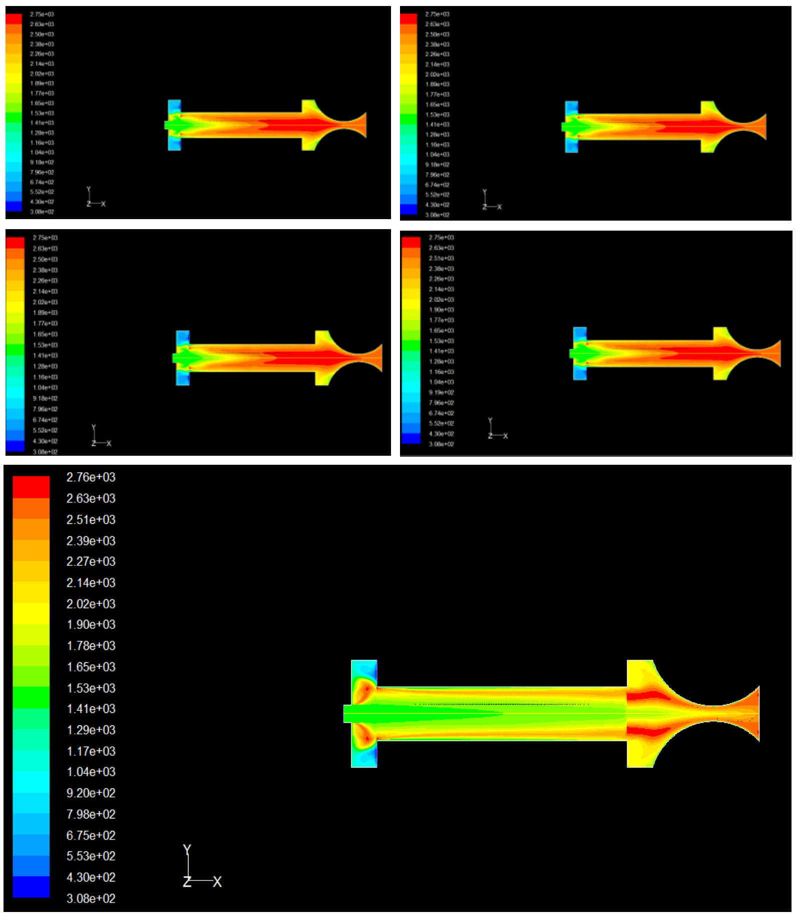

Figure 6

Contours of static temperature for 50,80,110,140 and 170m/s

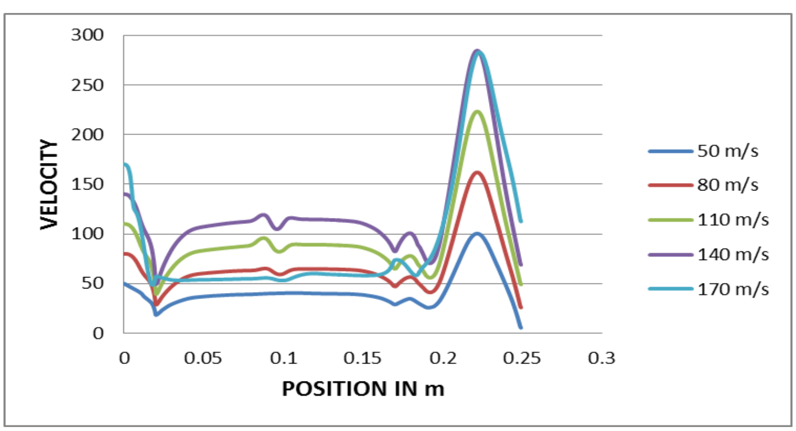

Figure 7

Velocity contoure along the position

The fuel-oxidizer mixture in this model is selected as Oxygen-Butadiene mixture. Oxygen is selected because the Solid Rocket motor is selected for the analysis. The PDF table is calculated with the respective fuel and oxidizer. Here the fuel is selected as Butadiene and the oxidizer is Oxygen.

3. RESULTS AND DISCUSSION

The analysis is carried out with steady flow and solver is pressure based. The viscous model is selected as k-epsilon with two equations. The Reynolds number of the flow is about 10^5. Thus, the flow is turbulent and the high-Re k-epsilon model is suitable. The species is considered as non-premixed combustion model (Figure 4).

The radiation of the model is assumed as P1 radiation because the P-1 model is one of the radiation models that can account for the exchange of radiation between gas and particulates. The material is selected as fluid for PDF mixture which is combusting particle for continuous phase and the wall is considered as solid which is considered as Aluminium. The PDF mixture is taken as ethylene-Oxygen (C_2H_4-O_2) mixture. The result is computed from the velocity inlet with the velocity of 50 m/s. and the temperature for the inlet is 1500 K. The wall is assigned as immovable. The furnace wall will be treated as an isothermal boundary with a temperature of 1200 K. The outlet is assigned as outflow and the temperature of the outlet is 2000 K. The solution is initialized from the velocity inlet. The kinematic energy **K=0.0001 m^2s^2** and the **dissipation epsilon=1000 m^2s^3.**

In this model the inlet velocity is given as 50 m/s. The wall has a constant temperature of 1200 K the inlet temperature is assumed as 1500 K and the outlet temperature is 2000 K. The Reynolds number based on the inlet dimension and the average inlet velocity is about 100,000. Thus, the flow is turbulent. Only half of the domain width is modeled because of symmetry. At first the steady flow is considered. The non-adiabatic system is considered. The mean temperature attains a peak value of 2750 K at MMF of 0.05 and decreases gradually and attains the 500 K mark at the MMF of value 1. The mean density increases gradually with the increase of MMF. The mean density becomes 1.6 at the MMF of 1.

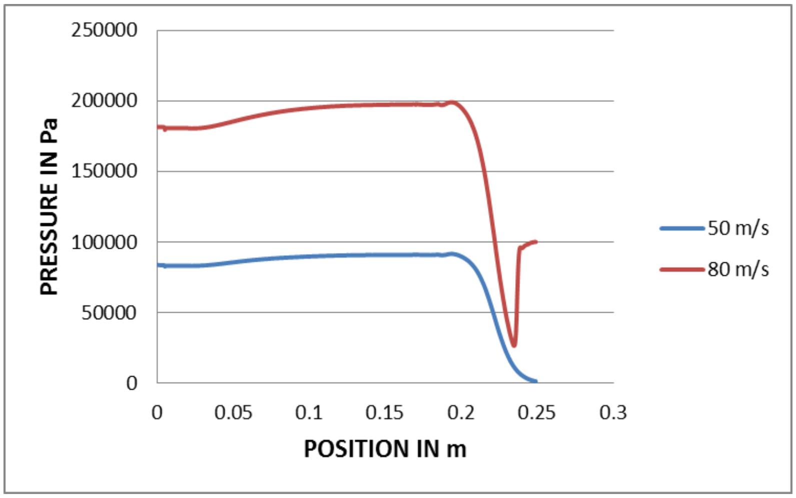

Figure 8

XY Plot of static pressure at 50 m/s and 80 m/s

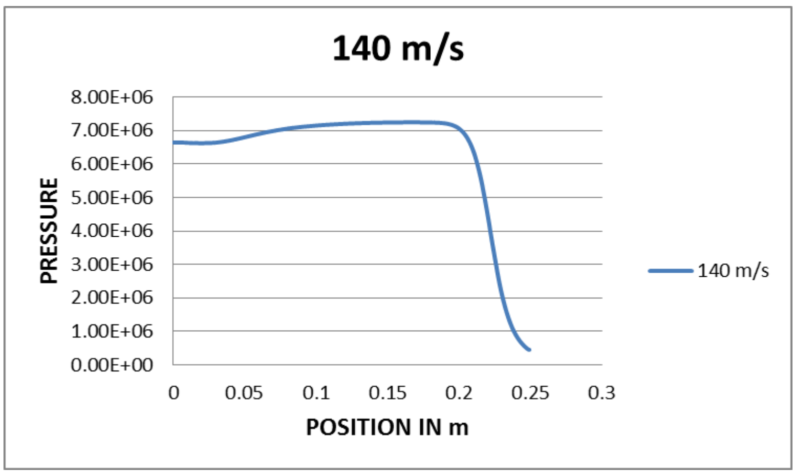

Figure 9

XY Plot of static pressure at 140 m/s

CO2 is formed after combustion and becomes maximum of 0.201 after the burning of all the fuel and oxygen and is represented in the contours. In the figure 5, the temperature at the inlet is given as 1500 K and the wall temperature is 1200 K and after the combustion, the temperature reaches the highest peak value of 2750 K in the combustion area and at the nozzle exit the temperature is 2630 K. In the figure 6, the temperature at the inlet is given as 1500 K and the wall temperature is 1200 K and after the combustion, the temperature reaches the highest peak value of 2750 K in the combustion area and at the nozzle exit the temperature is 2630 K. In figure 7 the temperature variation is same up to 140 m/s but the variation occurs in the 170 m/s because at high velocity, the combustion has the risk of blown out which can reduce the temperature. the high temperature is attained at exit area and the backward facing step. The peak temperature is 2760 K. The variation of temperature in the symmetrical region for the velocity of 50 m/s. The peak value of temperature is 2750 K.

The XY plots denoting the pressure variations of 50 m/s and 80 m/s is plotted in the same graph and is compared. The pressure variation is very high while using the solid rocket motor. This instability plays some major roles in the structural damages of the combustion chamber. They can be identified using the CFD and can be verified instead of using experimental setup which is time and economy consumable.

The pressure contours for the velocity inlet of 80 m/s is represented and the maximum pressure is 1.99 MPa and at the exit area is 26400 pa.

In the figure 8, the pressure contours for the velocity inlet of 140 m/s is represented and the maximum pressure is 2 MPa and at the exit area is 27900 pa. the pressure contours for the velocity inlet of 140 m/s is represented and the maximum pressure is 2 MPa and at the exit area is 27900 pa. The above contours are plotted in the graphs for representing the variation of the pressure. In figure 9, the pressure variance with the change of inlet velocity is plotted in the graphs. The varying velocities are 50 m/s, 80 m/s, 140 m/s and 170 m/s

4. CONCLUSION

The pressure variation is very high while using the solid rocket motor. This instability plays major role in the structural damages of the combustion chamber. They can be simulated using the CFD and can be verified instead of using experimental setup which is time and economy consumable.

REFERENCE

1. Abdul Raheem A. S. Numerical Simulations of Unsteady Flows in Solid Rocket Motors. AIAA Paper. 2004-2878.
2. Pandey, K.M. CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1. IJCEA, 2010, 1, 10-16.
3. Pardhasaradhi N, V. Ranjith K. and Rao. H. Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics. IJERA, 2012, 2, 35-41.
4. Sanjeev. M. On Combustion Instability in Solid Rocket Motors. California Institute of Technology Pasadena, California 2004.
5. Sarah L. Kulhanek. Design, Analysis, and Simulation of Rocket Propulsion System. University of Kansas, Lawrence, U.S.A.

6. Shanbhogue, S. J. Sujith, R. I., and Chakravarthy, S. R. Aero acoustics of rocket motors with FINOCYL grain. AIAA Paper, 39th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 2003-4632.
7. Vipul. S. CFD Analysis of Rocket Nozzle. University of Petroleum & Energy Studies, Dehradun, 2011.